Finite element method applied to plane frames

Filipe M. S. Oliveira[1], Felipe A. V. Bazán[1], Marina C. Micenas[1], Paulo C. O. Queiroz[1], George F. Azevedo[1]

[1]*Dept. of Civil Engineering, Federal University of Maranhão*
Av. dos Portugueses, 1966. Campus do Bacanga, 65080-805, Maranhão/São Luís, Brazil
filipe.oliveira@discente.ufma.br, felipe.vargas@ufma.br, marina.micenas@discente.ufma.br, pco.queiroz@ufma.br, gf.azevedo@ufma.br

Abstract. The calculation of displacements and forces in structural models is of great relevance in structural design and provides information about the behavior of the structure under different loading conditions. Due to the complexity of real problems and the demands of the production sector, the solution of a structural model becomes unfeasible without the aid of computational tools. The direct stiffness method, also known as matrix structural analysis, can be considered as a particular case of the Finite Element Method (FEM). This article presents a software for structural analysis of plane frames using the direct stiffness method. The software considers the presence of internal hinges and elastic supports in the structural model. The computational code was developed in Python language. Excel files are used as data input and output. Results include information such as nodal displacements, internal forces, and support reactions in the structure. Variations of internal forces along the frame axis, i.e. internal force diagrams, are presented in graphical manner. The influence of considering rigid or elastic supports on the results is investigated in the article.

Keywords: FEM, plane frames, Python.

1 Introduction

The structural analysis considers supports that limit movement and ensure the stability of the buildings and the use of the Finite Element Method (FEM) in various structural analysis software, such as ANSYS, ABAQUS, Ftool, among others, plays a key role in solving complex problems related to civil engineering. According to Martha [1], a structural model has boundary conditions in terms of displacements and rotations that represent the connections with the external environment, determined by supports that define the support conditions at the external contact points. Süssekind [2] classifies these supports into three types: The first type of support restricts displacement in the vertical direction, generating a force reaction in this direction while allowing free rotation and displacement perpendicular to the restricted direction. The second type of support restricts displacements in horizontal and vertical directions, generating two force reactions while allowing free rotation. The third type of support restricts all movements of the structure, generating force reactions in both horizontal and vertical directions, as well as a moment reaction around the support. The function of supports is to limit the degrees of freedom of a structure, restricting its movement tendencies and thus generating the conditions around the problem. According to Assan [3], in the Rayleigh-Ritz method, the elastic function is replaced by an approximate function, formed by the linear combination of other functions, known as shape functions. A third-degree polynomial is used as an approximating function [4], and through the boundary conditions, the shape functions are obtained. Then, the total potential function for the finite element of the frame is generated, and through the minimization of this potential, the stiffness matrix and the load vector for the finite element of the plane frame are found.

2 Description of algorithms

According to Vaz [5], the stiffness matrices and the load vector for the plane frame finite element, generated by the Rayleigh-Ritz method, refer to the local coordinate system, where the horizontal axis corresponds to the axis of the members. Thus, based on Alves Filho [6] these matrices must be transformed, by means of the rotation matrix, to the global coordinate system, horizontal and vertical axis of the Cartesian system. Every element's stiffness matrix must be added by term-to-term superposition, considering the row and column to which the matrix term refers, the same is done for the element's load vectors. This is how the stiffness matrix of the structure and the load vector of the structure are generated. The problem of structural analysis boils down to solving a matrix system. After the resolution of this system, making use of the boundary conditions of the model, the nodal displacements, and the vector of displacements of the structure are obtained. From there, the displacement vector of each element is constructed, and enough information is acquired to calculate the support reactions and the internal forces of the model.

3 Application and results

Martha [1] states that the main objective of structural analysis is to determine the internal and external forces (loads and support reactions), the corresponding stresses, as well as the determination of the displacements and the corresponding deformations of the structure to be designed. Thus, to perform the structural analysis of a plane frame, this work uses a program, implemented in the Python programming language, which uses a pre-organized Excel spreadsheet with the input data of the structure to be analyzed and, after processing, generates another Excel file with the output data, providing the required information to obtain the response of the structure to the applied loads. This includes nodal displacements, support reactions, and internal nodal forces in the structural model. The main purpose of the tool is to examine plane frames, which are common structures in civil engineering. These frames can be classified as isostatic or hyperstatic, having respectively reactions equal to or more than necessary to ensure stability. Thus, it is intended to present the results of the application of the algorithm, which applies the concept of matrix analysis, a version of the FEM, in the structural analysis of plane frames. The matrix analysis approach proposes an accurate and efficient method to deal with the essential calculations in structural analysis. The use of matrices allows the representation and solution of complex systems of equations, a fundamental aspect when using FEM. The specific choice of Python as a programming language demonstrates a concern with the accessibility and flexibility of the tool. The routine can also incorporate the use of internal hinges, elastic supports and elastic base, the latter being a way to contemplate the soil-structure interaction. In this context, Fig. 1 shows an example of a plane frame taken from Martha [1].

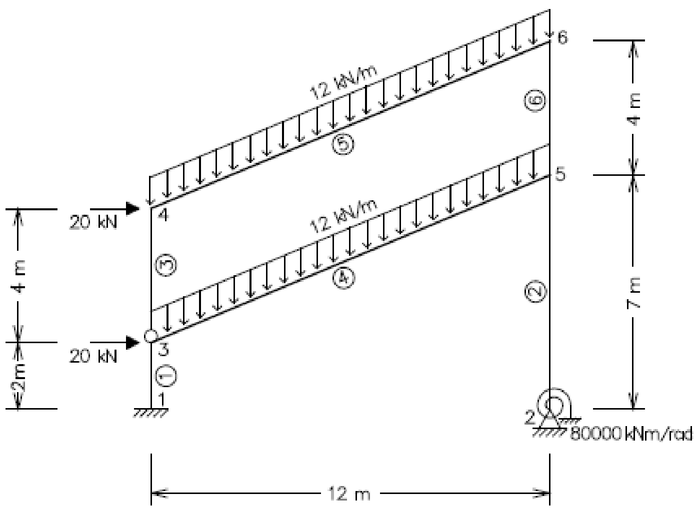

Figure 1 - Example of a plane frame [1]

For the structural analysis, the data presented in Tab. 1 and Tab. 2 were used as the input file. Thus, an analysis with a discretization of six nodes and six elements was carried out. The discretization of the structural model with the indication of nodes and elements is shown in Fig. 1.

Table 1. Input data for nodes

Node	Coordinate X (m)	Coordinate Y (m)	Constraint Horizontal	Constraint Vertical	Constraint Rotational	Applied loads Horizontal (kN)	Applied loads Vertical (kN)	Applied loads Moment (kN*m)	Elastic support Horizontal (kN/m)	Elastic support Vertical (kN/m)	Elastic support Rotational (kN*m/rad)
1	0	0	1	1	1	0	0	0	0	0	0
2	12	0	1	1	0	0	0	0	0	0	80000
3	0	2	0	0	0	20	0	0	0	0	0
4	0	6	0	0	0	20	0	0	0	0	0
5	12	7	0	0	0	0	0	0	0	0	0
6	12	11	0	0	0	0	0	0	0	0	0

Table 2. Input data for elements

Element	Initial node	Final node	Cross sectional area (m²)	Elastic modulus (kN/m²)	Moment of inertia (m⁴)	Axial distributed load (kN/m) Initial node	Axial distributed load (kN/m) Final node	Transverse distributed load (kN/m) Initial node	Transverse distributed load (kN/m) Final node	Internal Hinge Initial node	Internal Hinge Final node
1	1	3	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	0	0	0	0	0	0
2	2	5	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	0	0	0	0	0	0
3	3	4	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	0	0	0	0	1	0
4	3	5	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	-4.6154	-4.6154	-11.0769	-11.0769	0	0
5	4	6	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	-4.6154	-4.6154	-11.0769	-11.0769	0	0
6	5	6	8.0×10^{-3}	2.0×10^{8}	4.0×10^{-4}	0	0	0	0	0	0

After processing the data with the algorithm, an output file with the results presented in Tab 3 to 5 was exported:

Table 3 - Support reactions

Node	Horizontal reaction (kN)	Vertical reaction (kN)	Moment-reaction (kN*m)
1	-23.6	158.0	144.2
2	-16.4	154.0	39.4

Table 4 - Nodal displacements

Node	Horizontal displacement (m)	Vertical displacement (m)	Rotation (rad)
1	0	0	0
2	0	0	-4.929×10^{-4}
3	3.212×10^{-3}	-1.975×10^{-4}	-3.015×10^{-3}
4	1.482×10^{-3}	-4.507×10^{-4}	-1.842×10^{-3}
5	3.789×10^{-3}	-6.739×10^{-4}	1.087×10^{-3}
6	1.114×10^{-3}	-8.107×10^{-4}	2.054×10^{-3}

Table 5 - Internal forces

Element	Normal force (kN)		Shear force (kN)		Bending moment (kN*m)	
	Initial node	Final node	Initial node	Final node	Initial node	Final node
1	-158.0	-158.0	23.6	23.6	-144.2	-97.0
2	-154.0	-154.0	16.4	16.4	-39.4	75.6
3	-101.3	-101.3	-34.1	-34.1	0	-136.5
4	13.0	73.0	66.8	-77.2	-97.0	-164.5
5	-88.9	-28.9	72.7	-71.3	-136.5	-127.6
6	-54.7	-54.7	54.1	54.1	-88.9	127.6

Subsequently, normal force, shear force and bending moment diagrams for the structure are plotted in Fig. 2 to Fig. 4:

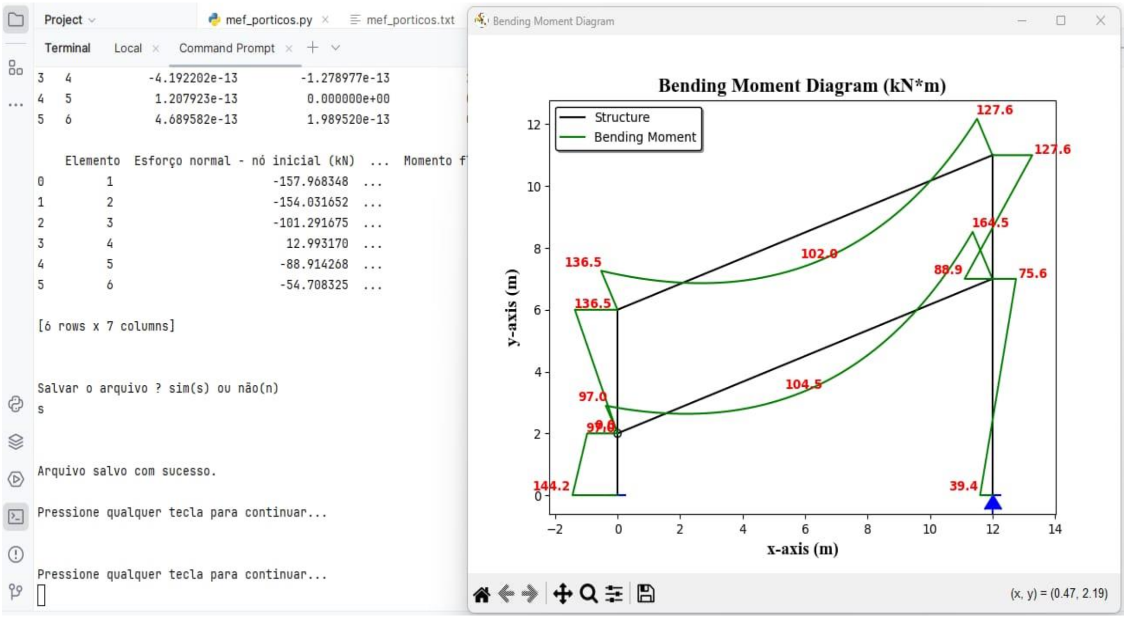

Figure 2 – Bending moment diagram

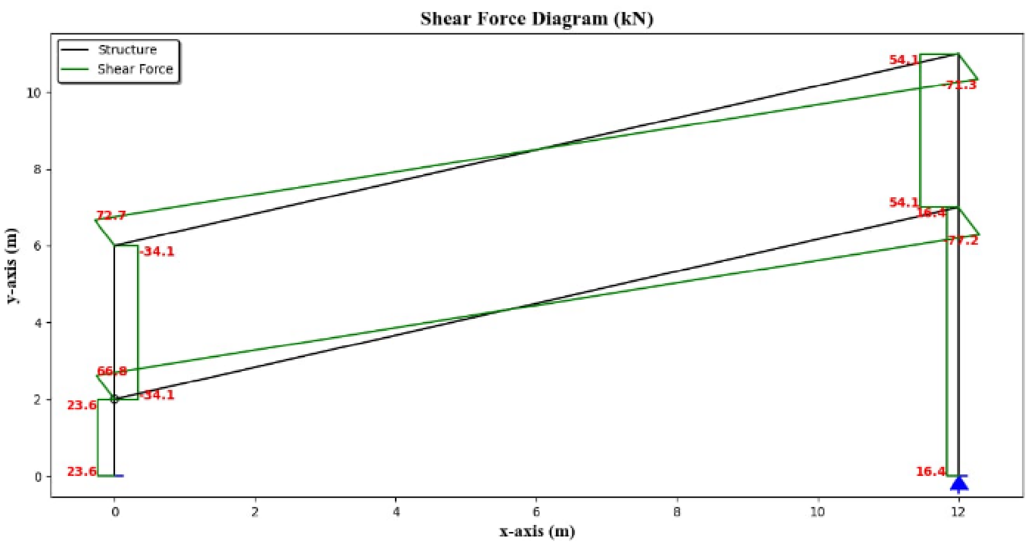

Figure 3 - Shear force diagram

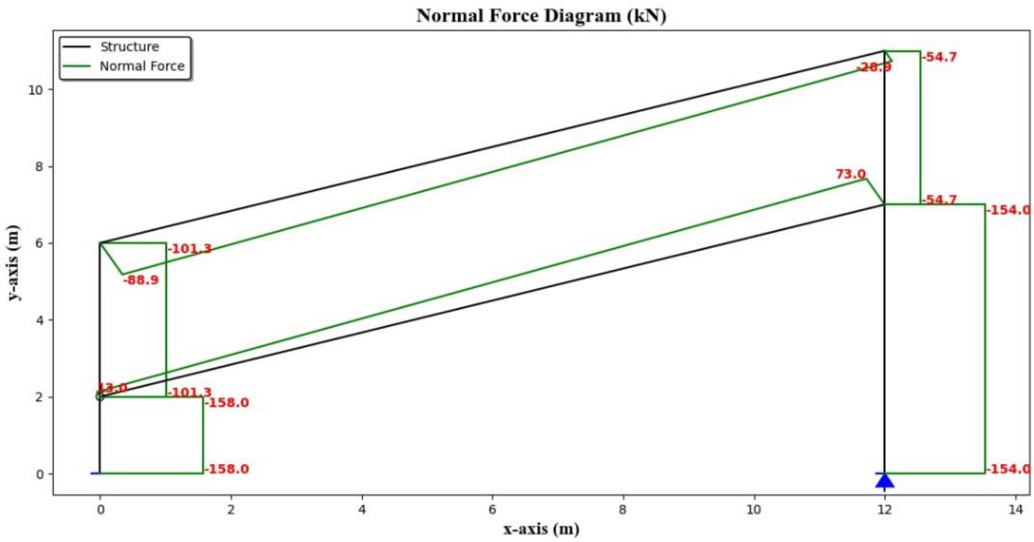

Figure 4 – Normal force diagram

3.1 Validation

All the results of the example plane frame shown in Fig. 1, including nodal displacements, support reactions, and internal forces, are identical to the ones presented by Martha [1].

4 Conclusions

According to the data obtained through the application and validation of the results, it is understood that the program can solve complex structural analysis problems and plot the diagrams, thus facilitating the visualization of the internal forces and the determination of the structure behavior in response to the imposed loads and constraints.

Acknowledgements. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors or has the permission of the owners to be included here.

References

[1] L. F. Martha. *Análise de estruturas: conceitos e métodos básicos*. Elsevier, 2010.
[2] J. C. Süssekind. *Curso de análise estrutural: estruturas isostáticas*. Globo, 1981.
[3] A. E. Assan. *Método dos elementos finitos: primeiros passos*. Editora da Unicamp, 2003.
[4] J. B. Paiva. *Introdução ao método dos elementos finitos: notas de aula*. EESC/Universidade de São Paulo, 2012.
[5] L. E. Vaz. *Método dos elementos finitos em análise de estruturas.* Elsevier, 2011.
[6] A. Alves Filho. *Elementos finitos: a base da tecnologia CAE.* Érica, 2013.